Composite Lattice Reinforced Part Optimization with FEA: An Automotive Door Component Case Study

Meghana Kamble[1], Robert Jopson [2], Jeffrey Wollschlager [2], Christopher Oberste[1]
[1] WEAV3D Inc, Norcross, Georgia, USA
[2] Altair Engineering, Inc, Troy, Michigan, USA

ABSTRACT

This paper presents a finite element analysis (FEA) method of modelling automotive components reinforced with WEAV3D composite lattice structures. WEAV3D Inc. has developed a novel hybrid-material approach, Rebar for Plastics®, merging existing thermoforming, compression molding or injection overmolding processes with woven composite lattices formed from unidirectional thermoplastic tapes to produce high-rate, cost-effective structural composites. These materials pose challenges for conventional composite FEA modeling, demanding advanced techniques for accurate mechanical predictions, especially in structural applications.

Presented FEA methodology addresses these challenges by enabling explicit modeling of lattice designs using Altair HyperMesh. By parameterizing tape materials, spacing, and layer count through scripting, the methodology streamlines design exploration, eliminating the need for CAD models for each iteration and enabling regional optimization within the part. This optimizes lattice structures efficiently to achieve desired mechanical properties at the part level.

Through a case study of an automotive door component, we demonstrate the applicability of our FEA workflow. Comparison of FEA predictions with experimental data validates the reliability of the model. These results highlight the efficacy of our methodology in predicting mechanical behavior, suggesting its potential to guide design decisions across various industrial applications through optimization.

1. INTRODUCTION

Recent advancements in thermoplastic composite manufacturing have led to the creation of hybrid-molded composite structures that integrate continuous fiber composites with injection or compression molded compounds. This capability allows engineers to produce structural parts with selective reinforcement while still maintaining high production rates [1]. However, integrating continuous and discontinuous fiber-reinforced thermoplastics poses challenges, especially in accurately predicting mechanical behavior using traditional Finite Element Analysis (FEA) models, which often focus on single-layer homogeneity. Traditional FEA techniques, which are tailored for isotropic materials and ply-based composites, fall short in accurately representing the complex physical properties and behaviors of these hybrid structures.

This inadequacy is particularly pronounced when attempting to model multi-material composites having heterogenous material distributions over the surface of the part. WEAV3D Inc. has developed an innovative approach, Rebar for Plastics®, combining existing thermoforming, compression molding, or injection overmolding processes with woven composite lattices made from unidirectional thermoplastic prepreg tapes, enabling designers to achieve the required modulus, strength, and impact properties of the final structural component by adjusting the lattice design by varying the tape material and spacing between the tapes (weave density) in a lattice layer(ply), as shown in Figure 1. The flexibility to alter weave density and fabric material within a single layer significantly expands the design space, necessitating a workflow capable of managing multiple simultaneous optimization variables.

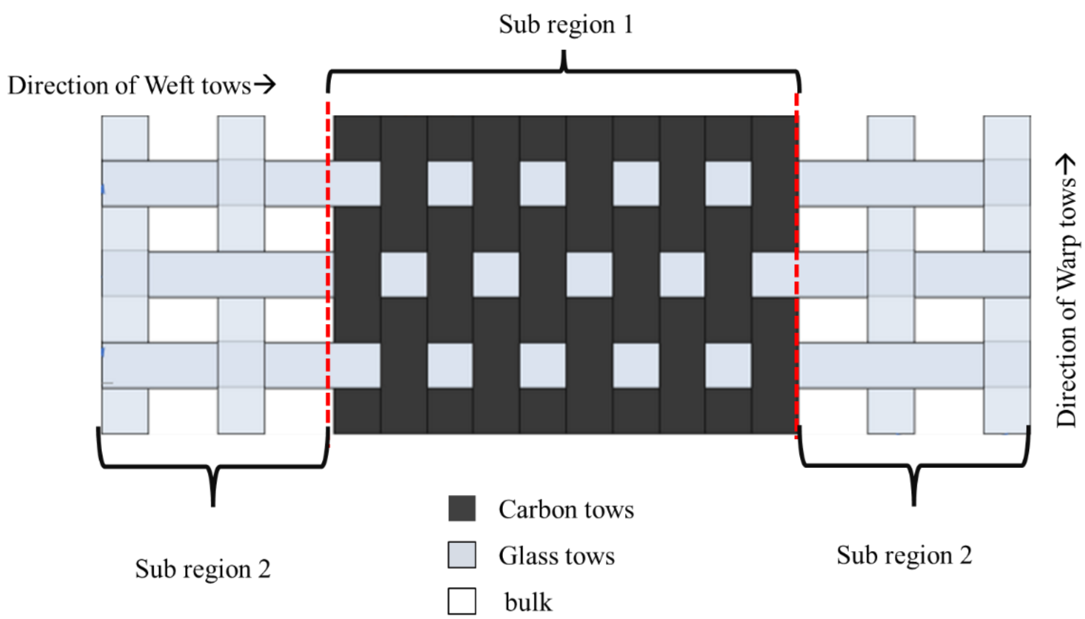


Figure 1: Example of a Heterogeneous Lattice with Variable Tape Spacing and Tape Material

As a result of the existing limitation, WEAV3D and Altair were motivated to develop a new, robust tool to address the specific simulation challenges posed by WEAV3D's Rebar for Plastics® technology. Our methodology provides a validated workflow for the efficient and reliable design of high-performance hybrid-molded structural parts.

2. BACKGROUND

Several notable analytical models exist to describe the elastic stiffness of woven-fabric reinforced composites, with the majority relying on decomposition individual plies into representative volume elements (RVEs) that capture a repeating unit of the fabric. Where these models tend to differ is on the degree of simplification that is applied to the RVE, as simpler models are easier to calculate but tend towards higher predictive error. The fabric geometry model (FGM) is frequently used to model the elastic behavior of woven-fabric reinforced composites without requiring extensive computational resources [2]. The FGM treats the fiber and matrix in the composite as a collection of composite rods within an RVE and uses a stiffness-averaging method to generate a global stiffness tensor for the composite by averaging the local stiffness tensors of each rod, weighted by their relative volume fractions within the RVE. Because the local stiffness tensors are averaged to create the global stiffness tensor, modeling heterogeneous, repeating fabric geometries is as straightforward as modeling homogeneous ones.

While the FGM model can reliably predict the stiffness of woven-fabric reinforced composites, it also requires the user to completely characterize the crimp effects and fiber volume fraction of each composite rod in the RVE, which can be difficult to achieve in practice. A simpler method, the mosaic model, opts to ignore crimp effects entirely by converting the fabric into planar tiles. According to Ishikawa and Chou, "this model is idealized as an assemblage of asymmetrical cross-ply laminates" [3]. Using classical laminated plate theory, the stiffness of these planar tiles can be calculated for the smallest repeat unit of the composite, which is considered representative of the overall composite stiffness (Figure 2). Additionally, a one-dimensional model provides an approximate solution for determining the mechanical properties of hybrid composites or composites containing multiple types of reinforcement fibers.

Finite element analysis (FEA) is considered the most accurate method of modelling composite properties; however, it often struggles to simultaneous model micro and meso-scale behavior due to meshing limitations. In order to resolve this issue, we had previously developed an RVE-based approach using ANSYS 2021 R1 - Material Designer. Modeling the RVE in this method requires creating simplified 3D geometry of the RVE unit cell and defining the material properties of the constituent materials. The simplified RVE assumes that the weft and warp tapes sit on top of each other rather than replicating the actual interlacing of woven material, much like the mosaic model. In our case, this assumption has only a small effect on the final homogenized properties, as the crimp angle ranges from 0.8 to 2 degrees due to the small thickness of the prepreg tapes (0.15-0.3 mm) [4]. The constituent materials include the lattice tapes (weft and warp) and the overmolded plastic between the tapes in a single lattice layer.

The RVE geometry is then meshed for FEA. ANSYS subjects the RVE to various macroscopic load cases and computes the response. From these responses, the homogenized orthotropic material data for a single layer of the lattice is derived. Each RVE is then assembled into a laminate, equivalent to the planar tiles described by Ishikawa and Chou, to define the thickness of the part and the stiffness of these planar tiles, calculated using classical laminate plate theory, represents the stiffness of the overall composite.

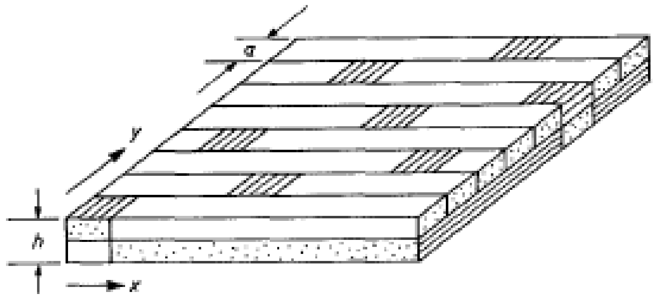


Figure 2: Representative element of the mosaic model

In a heterogenous lattice like shown in Figure 1 material properties change within a ply. The designer must split the ply into small sub-regions (Figure 3) with constant material properties, creating an RVE for each sub-region in Material Designer. This necessitates a library of such RVEs, where each RVE can be meshed and solved to obtain homogenized orthotropic material properties. These properties are then assigned to their respective sub-regions based on the allocated RVE. While this method enables the generation of a single ply with varying material properties in ANSYS ACP and can be used for complex geometries, this RVE based workflow has a notable limitation: it cannot parametrize the spacing of unidirectional tapes within a lattice geometry, necessitating a new RVE CAD for every iteration. Additionally, a post processing submodeling step is required to differentiate between the stresses of the tape and the bulk and also requires the designer to accurately predict the tape locations within the submodel, which becomes challenging for complex geometries.

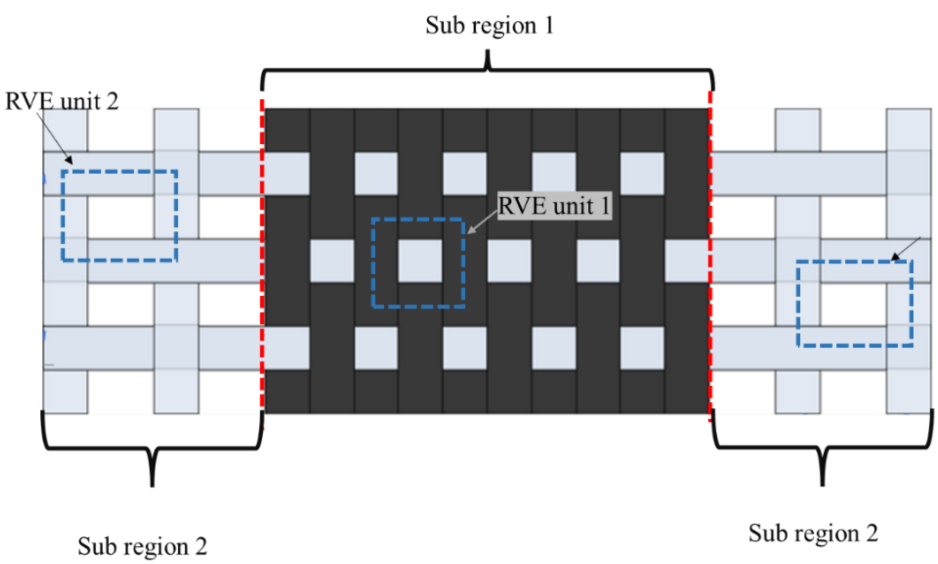


Figure 3: Ply split into sub regions for RVE allocation

Explicit FEA modeling, used in this context to describe a part level model whereby the reinforcing material is physically modelled within the meshed geometry, has the potential to resolve a number of these issues and has been used to good effect in other applications, such as the modelling of steel rebar reinforced concrete. Unlike RVE methods, which implicitly represent the reinforcement via smearing of properties throughout the ply, an explicit approach keeps the reinforcement and bulk material (in our case, a molded plastic material) distinct throughout the simulation process, which allows for accurate extraction of stress data from each constituent component.

3. METHODOLOGY

The explicit FEA approach developed by WEAV3D, and Altair assigns elements specific tow or bulk properties and automating the process through scripting. This method simplifies the management of various design variables, including tow materials, number of layers, and tow spacing. With explicit FEA, only a part-level CAD model is required, which is meshed and solved for FEA analysis. Preprocessing involves creating material cards and defining lattice design properties in an input text file. The FEA analysis is then solved, and deformation and stress results in the tows and bulk layers are obtained in a single post-processing step. This automation enhances efficiency, reduces manual intervention, and allows for a more flexible and comprehensive exploration of design configurations.

The core of this explicit approach is a Tcl script that leverages the Altair HyperMesh API. We have previously published a detailed guide to this methodology at CAMX 2023 [5], however, a summary of the script operation steps are as follows:

1. Parsing Tow Definitions
2. Geometry Meshing and Material Definition
3. Tow Orientation and Projection
4. Element Organization
5. Bulk Layer Definition
6. Composite Structure Formation
7. Shape Adaptability

The script starts by parsing tow definitions from a text file, creating tow objects with direction, width, thickness, material, and spacing attributes. It then meshes the geometry and establishes a local coordinate system, defining material properties for tows and the bulk layer in the material database. The script guides the user through prompts to define the component or sub region where the tows will be incorporated. Using the local coordinate system, tows are oriented and projected onto the part geometry along the z-axis, with an option for symmetry to enable lattice on one or both sides of the component. In the example shown in Figure 4, there is a single lattice region covering the entire panel surface and the design exhibits symmetry across the thru thickness midplane (i.e. the panel is configured as a sandwich structure) with bulk plastic positioned between top and bottom lattices. Elements within each tow's projection are organized into tow objects, creating ply entities in warp and weft directions (Figure 5). The bulk layer is defined with a final ply entity, calculating bulk thickness as the difference between the total part thickness and the cumulative thickness of intersecting tows. This variable thickness is assigned via a table. Ply entities are then stacked within a laminate entity to form the composite structure as shown in Figure 4. The script handles tows for flat and slightly curved shapes, with more advanced projection algorithms required for complex geometries with sharp radii or significant changes in the direction of the surface.

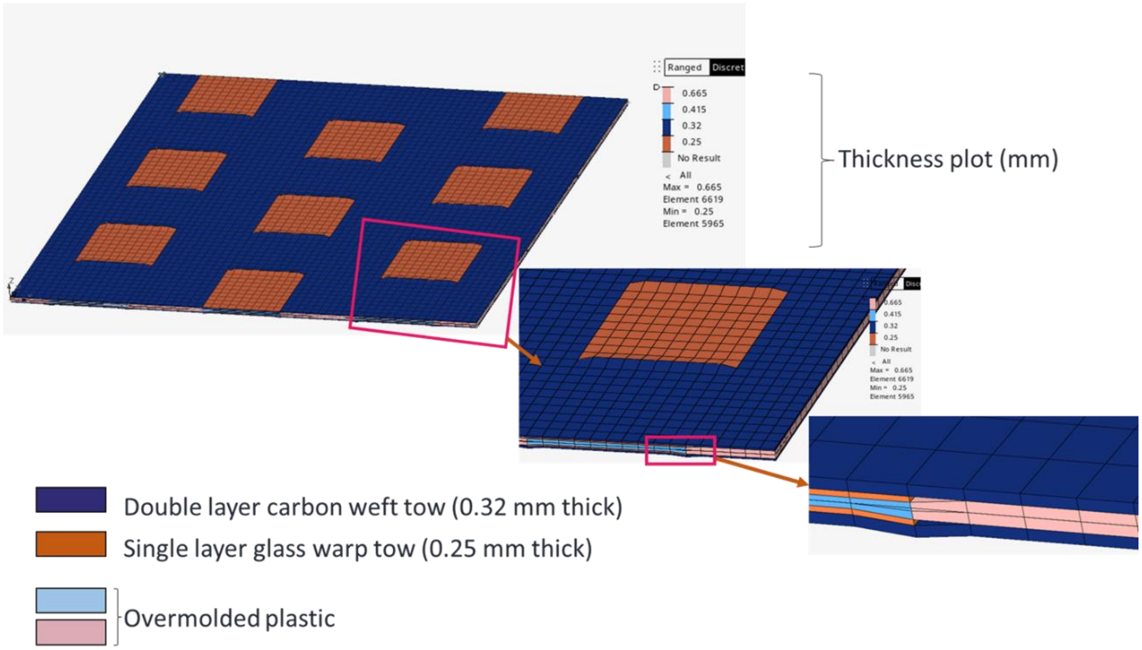


Figure 4 : Tow generated In Altair HyperMesh (Thickness plot)

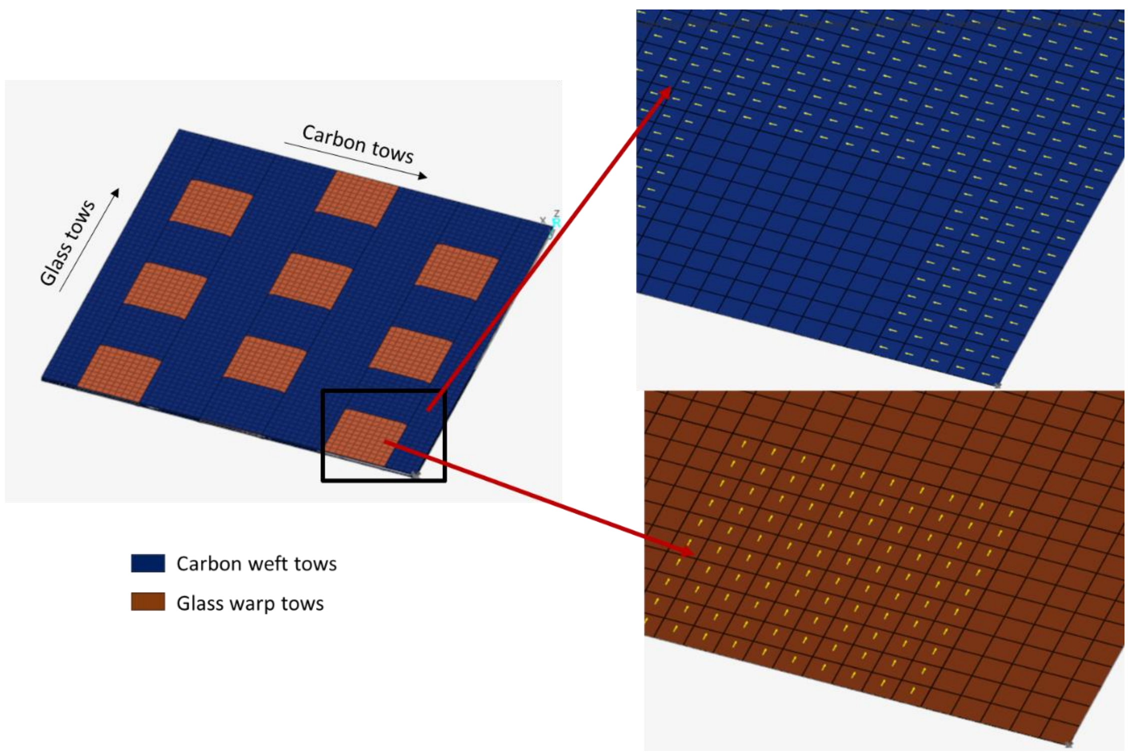


Figure 5: Material direction of tows generated in Altair Hypermesh

4. FEA EXPLICIT MODEL (THREE-POINT BEND TEST)

4.1. Experimental Setup

Plaques were manufactured for flexural testing (experimental setup shown in Figure 6) to validate the presented Explicit Model approach using Altair HyperWorks. The plaques had dimensions of 152.4 x 152.4 mm and a nominal thickness of 2 mm. Four plaques were produced for each of the four configurations, as summarized in Table 1.

The lattice patterns were created using 25.4 mm wide carbon and glass tapes with respective thicknesses of 0.16 mm and 0.25 mm. All designs utilized GF/PP warp tapes, 1 layer, 50.8 mm center-to center spacing. Each design included a lattice on both sides of an unfilled polypropylene sheet. All plaques were compression molded to cohesively laminate the layers together, with the weft tapes oriented along the primary flexure load direction. The plaques were loaded to failure, or to maximum deflection allowed by the fixture, with crosshead force and displacement data collected during the test. The flexure span was 63.5 mm.

Table 1: Lattice Design Configuration for Flexure Tests

Design No.	Molded Plastic Material	Weft Tow Material	Weft Tow		No. of Lattice layers
			No. of layers	Spacing (mm)	
1	Braskem Ti4003F PP	Glass/PP (45 % Vf)	2	25.4	2
2		Carbon /PP (40 % Vf)	2	50.8	2
3			2	25.4	2
4		Mixed -Alternating Glass/PP (45 % Vf) & Carbon /PP (40 % Vf)	2	25.4	2

Figure 6: Flexure Test Experimental

4.2. Experimental Vs FEA Flexure Test Results

For each of the designs described above, the chord modulus for each sample was calculated according to a modified version of ASTM D790 that accounts for the wider specimen dimensions. The average chord modulus for each design is reported as the experimental chord modulus in Table 2. The predicted FEA modulus was calculated formulaically based on the applied load and simulated displacements, with percentage error between the experimental and Explicit method derived chord modulus calculated to determine degree of correlation. Our flexure test FEA predictions, summarized in Table 2, show that the Altair's Explicit model exhibited good correlation with the experimental results, overpredicting the experimental modulus by an average of 5.8 % (0.3 % - 13.5 %).

Table 2 : Flexure Test - Chord Modulus Comparison for Lattice Integrated Panels

Design No.	Altair Chord Modulus (GPa)	Experimental Chord Modulus (GPa)	% Deviation
			Altair vs. Experiment
Design 1	25.98	25.64	1.32
Design 2	27.23	25.19	8.09
Design 3	53.14	52.99	0.28
Design 4	44.3	39	13.5

4.3. Methodology Comparison: User Experience

The ANSYS RVE method for lattice design involves four main steps:

1. RVE Development: Creation of RVE model, obtaining homogenized material properties.
2. Component Thickness Construction: Constructing the main component model.
3. FEA Analysis: Performing analysis to compute deformation and stress regions.
4. Submodeling: Detailed stress analysis within the tows and bulk layer.

Each step requires separate CAD models for the RVE, main component, and submodel. The preprocessing includes material data card creation, meshing, and boundary condition application. The ANSYS RVE method effectively addresses limitations of traditional ply-based composites FEA but is labor-intensive.

The Altair Explicit method simplifies this workflow:

- Requires only one CAD model.

- Preprocessing involves creating material cards and defining lattice design properties in a text file.

- FEA analysis is performed in a single step, yielding deformation and stress results for tows and bulk layers.

Compared to the ANSYS RVE method, Altair Explicit is over 50% faster in flexure tests with integrated lattice structures as summarized in Table 3. This efficiency reduces analysis time and supports iterative lattice design optimizations, making it a time-efficient solution for structural analyses.

Table 3: Comparison of ANSYS FEA Vs Altair Explicit methodology - Time to Setup and Solve - Flexural Load Case Applies to Flat Panel

ANSYS RVE Method			**Altair Explicit Method**		
Overview of Steps	**Setup Time**	**Solve Time**	**Overview of Steps**	**Setup Time**	**Solve Time**
RVE CAD	~ 3 minutes	~ 1 minute	*Input Text File*	~1 minute	N/A
ANSYS Pre	~5 minutes	~ 30 seconds	*FEA of the actual part*	~ 8 minutes	~ 40 seconds
FEA of Full Part	~ 5 minutes	~ 1 minute			
Submodel	~7 minutes	~ 1 minute			
Total Setup and Solve Time ~23 minutes			*Total Setup and Solve Time* ~10 minutes		

5. OPTIMIZATION PROCESS FOR DEVELOPING LATTICE REINFORCEMENTS

5.1. Overview of the Optimization Process

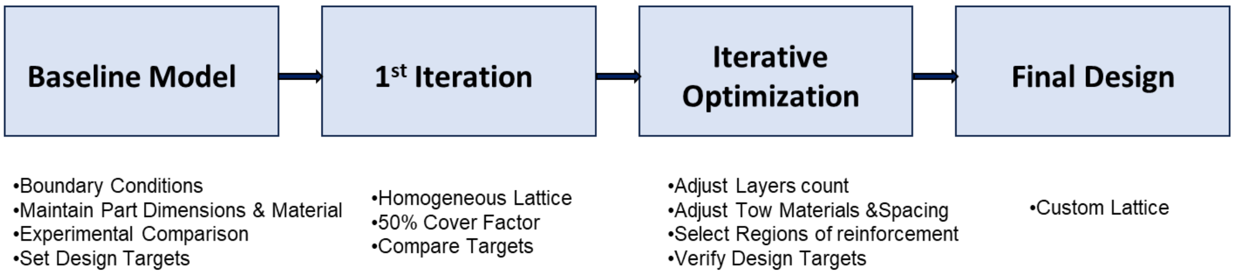


Figure 7: Optimization Process - Overview

As shown in Figure 7, the optimization process begins with the creation of a baseline FEA model. The primary objective of this initial phase is to develop an FEA model with boundary conditions that accurately replicate real-life testing scenarios. This involves using the part's original dimensions and baseline material properties. The results of the baseline FEA model, including stress and deflection plots, are then compared against actual experimental data to ensure alignment. Once the baseline FEA model is validated, design targets for optimization are established.

The optimization process typically starts with a review of the stress and deformation plots from the baseline model to assess its initial performance. For instance, consider an automotive component made from Natural Fibre Polypropylene (NFPP) nonwoven mats with a mass per unit area of 1000 gsm, which initially shows a y-axis deflection of 9.7 mm shown in Figure 8. Suppose the design goal is to reduce this deflection to 8.6 mm to meet a specified part performance requirement. Cost the primary optimization variable and weight is the secondary optimization variable.

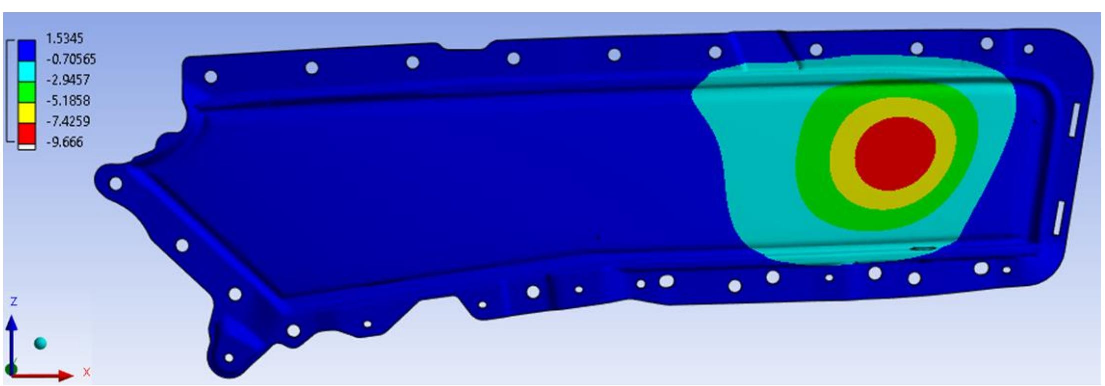


Figure 8: Example Deflection Plot of a door insert made of NFPP 1000 gsm.

To achieve this reduction, the designer might start by reinforcing the entire part with a single-layer homogeneous glass lattice featuring a 50% cover factor for the first iteration. The cover factor indicates the percentage of the area covered by the tape material in a specified dimension. In this example, after implementing the lattice reinforcement in the first iteration, the deflection reduced to 7.9 mm, successfully meeting the design target (Figure 9); however, the design is not optimized for cost or weight at this stage.

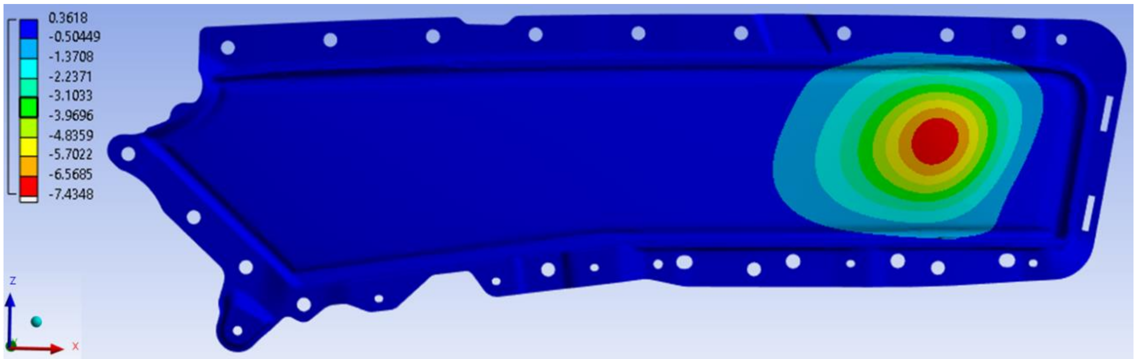


Figure 9: Deflection Plot of a NFPP 1000 gsm door insert reinforced with 50% lattice reinforcement.

To further optimize the design, the designer can review the initial von Mises stress plot to identify high stress and low stress regions i.e., design critical and non-critical areas (Figure 10). By selectively reinforcing only the high stressed regions with the lattice design chosen in the first iteration, the designer can refine the optimization process and compare the results against the established targets, which in this case is a deflection target. Selective reinforcement of the part is illustrated in Figure 11.

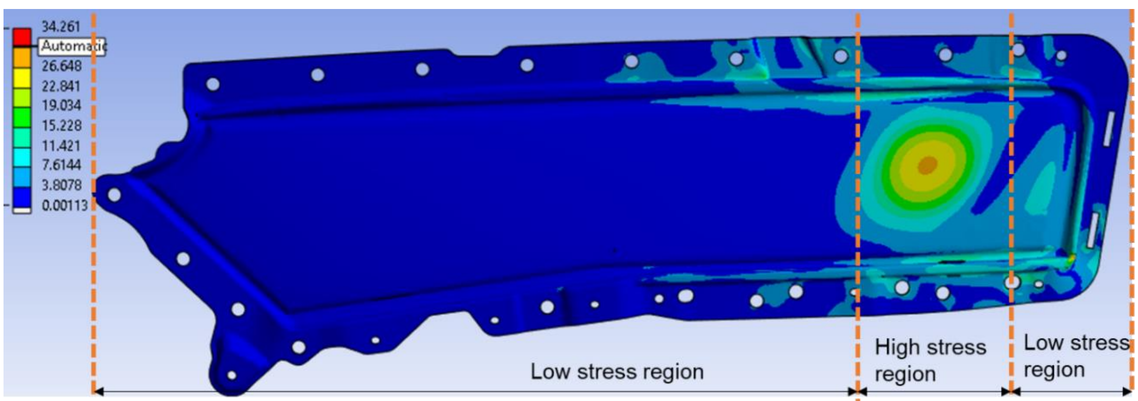


Figure 10 : Von Mises Plot - Identifying high and low stress region in the part.

Low stress region	Unreinforced area – NFPP 1000 gsm only
High stress region	NFPP reinforced with 50% lattice design

Figure 11 : Reinforcing the part with lattice in a high stress region only (mockup).

While this example scenario only required reducing deflection by 1mm, in scenarios requiring more significant deflection reduction or where stresses exceed the base material's yield stress, the designer might begin with a denser lattice configuration, such as a 100% cover factor, and then adjust the lattice density or layer count as necessary.

Subsequent iterations might involve developing heterogeneous lattice designs by adjusting the number of lattice layers, tow materials, and tow spacing. Additionally, the designer may select specific regions for lattice reinforcement and modify part geometry, such as adding ribbing or changing part thickness, to meet the design targets. All adjustments must respect manufacturing constraints. These iterations aim to achieve design goals related to strength, stiffness, cost, and weight, ultimately yielding a customized lattice design for the part by the end of the process

6. CASE STUDY: OPTIMIZING LATTICE DESIGN FOR AN AUTOMOTIVE PART

6.1. Baseline Model Validation

The NFPP automotive component shown in the prior section was selected for optimization in partnership with Antolin, one of the world's largest manufacturers of vehicle components and a global supplier of technology solutions for automotive interiors. We now present an optimization case study involving this interior door insert, originally manufactured using an NFPP nonwoven mat having a fiber weight fraction of 50%. The NFPP door insert (1700 gsm, 1.8 mm thickness) was subjected to a 60mm diameter, 150N load at the circular regions marked as points 8, 9, and 10, applied normal to the surface one at a time. In order to meet assembly-level performance requirements, the component level deflection at these points cannot exceed 8.6 mm. The door insert is attached to the rest of the assembly using fixation points, as shown in the Figure 12. These load cases and boundary conditions are translated into a baseline FEA setup. The fixation points are identified and fixed in all directions in the FEA. A load is applied over a circular region with a diameter of 60 mm along the Y-axis, normal to the surface. The baseline FEA deflections at points 8, 9, and 10 are compared against experimental values (Table 4), illustrating good correlation with the experimental results (-7% to +5%).

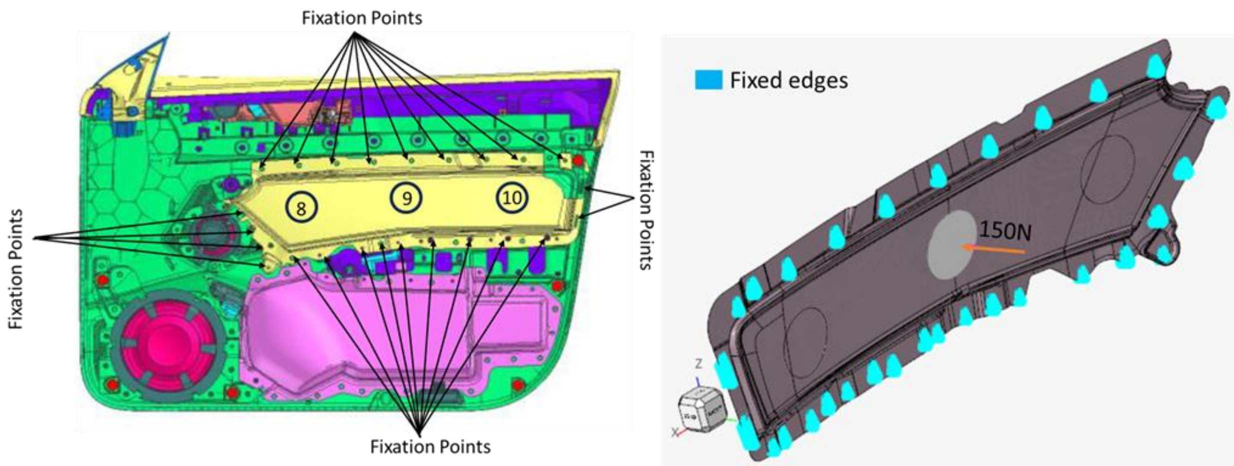


Figure 12: Door Insert (Left): Experimental setup; (Right): FEA Setup

Table 4 : Comparison of Baseline FEA with Experimental results (NFPP 1700 gsm)

Load Location	**Experimental Deflection (1700 gsm NFPP) (mm)**	**FEA Deflection (1700 gsm NFPP) (mm)**	**% Deviation**
			FEA vs. Experiment
8	7.48	7.7	3%
9	8.21	8.6	5%
10	6.35	5.9	-7%

6.2. Design targets for Optimization:

Once the baseline model validation was achieved, we established a design objective to demonstrate cost and weight neutrality or reduction relative to the baseline while ensuring that Y-axis deflection at all points (8, 9, and 10) remained below 8.6 mm. Weight and cost savings were to be achieved by utilizing a thinner NFPP mat (1200 gsm or 1000 gsm), with WEAV3D's composite lattice providing additional strength and stiffness as it was known that door inserts made solely of NFPP 1200 gsm or NFPP 1000 gsm exceeded the maximum deflection limit and failed (deflection results summarized in Table 5).

Table 5 : Deflections measured at locations 8, 9, &10 for NFPP only door inserts.

Door Insert Material	Deflection at Pt. 8	Deflection at Pt. 9	Deflection at Pt. 10
NFPP 1700 gsm	7.7	8.6	5.90
NFPP 1200 gsm	13.66	14.35	8.62
NFPP 1000 gsm	15.18	16.02	9.67

6.3. 1st Iteration

The baseline model deflection results indicated that the deflection at the observed points exceeded the allowable limit of 8.6 mm by 12% to 86%. To address this, a balanced approach was chosen for the first iteration, incorporating a single layer of glass fiber/polypropylene (GFPP) tow, 0.25 mm thick, with a 50% cover factor across the entire part. This approach was applied to door inserts fabricated from both NFPP 1200 gsm and NFPP 1000 gsm.

The results from the first iteration (Figure 13) indicate that the deflections for the homogeneous lattice reinforced NFPP 1200 gsm insert remained within the allowable limit of 8.6 mm; however, the homogeneous lattice-reinforced NFPP 1000 gsm insert slightly exceeded the allowable deflection limit at location 8. This provided a starting point for cost and weight optimization.

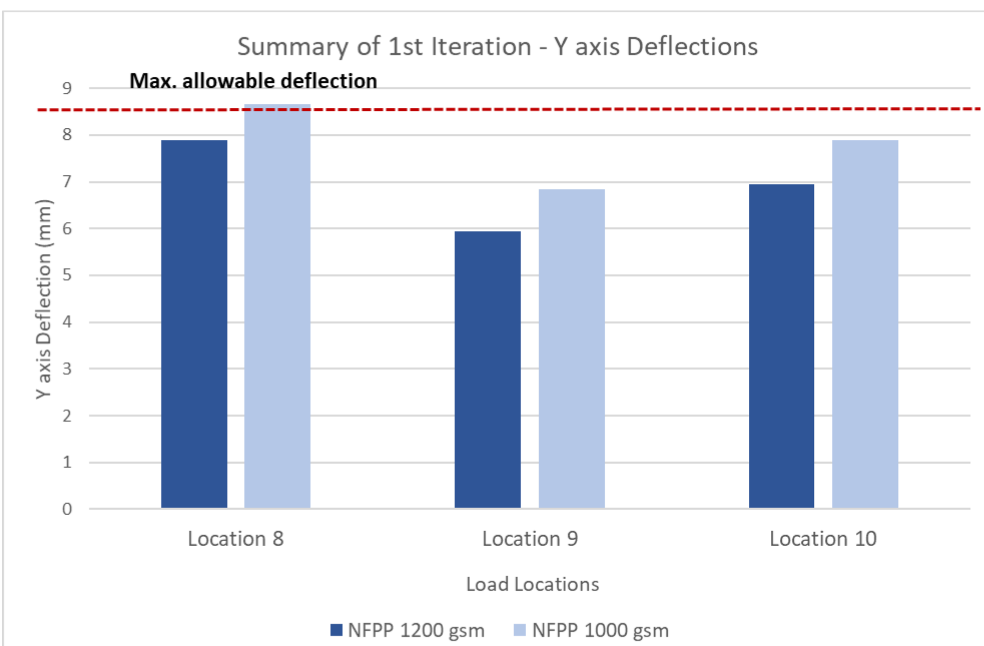


Figure 13: Y-axis deflection(mm) at different locations w.r.t to the target deflection

6.4. Iterative optimization

Figure 14 presents a collage of stress plots corresponding to points 8, 9, and 10 from the baseline FEA model. These plots reveal that the maximum stresses are primarily concentrated around the load application area, which has a diameter of 60 mm. Based on this analysis, it can be inferred that localizing the lattice reinforcement within the central area of the part should be sufficient to achieve the performance target and reduce material utilization; therefore, the part was divided into lattice and non-lattice regions.

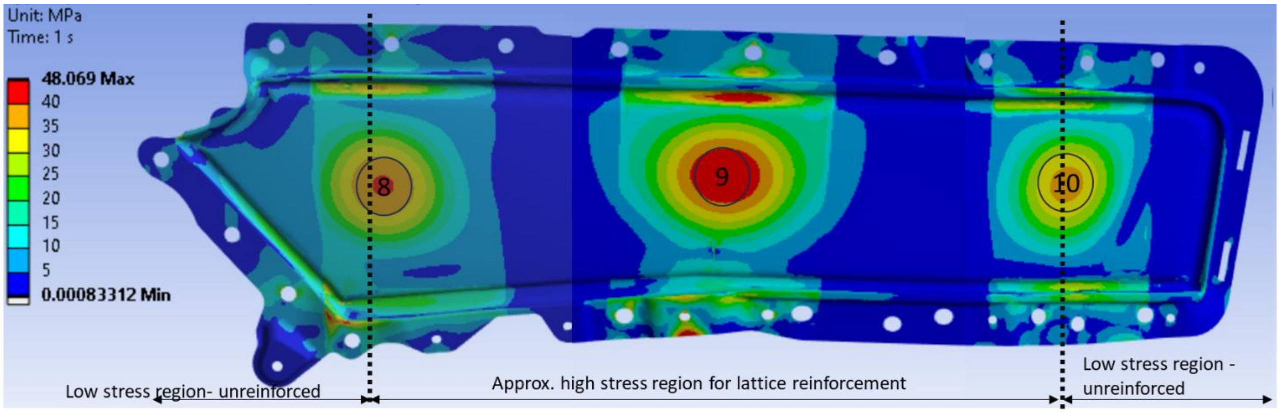


Figure 14: Combined Von Mises Plot of high stress regions around points 8,9 & 10

The optimization strategy focused on regions with high deflection, specifically those approaching the allowable deflection limit, which were assigned a denser lattice design. Conversely, the weave density was reduced in regions where deflections were not near the maximum limit. As a result, the identified lattice reinforcement region was further divided into three heterogenous sub-regions, illustrated in Figure 15, namely regions 8, 9, and 10, based on the stress distribution observed in the baseline model. Each region was assigned a unique weft cover factor. The weft tapes, which run vertically along the part from top to bottom, serve as the primary load-carrying elements. Meanwhile, the warp tows maintained a constant cover factor of 50% as we require at least 2 tapes to ensure stability of the lattice during handling and forming.

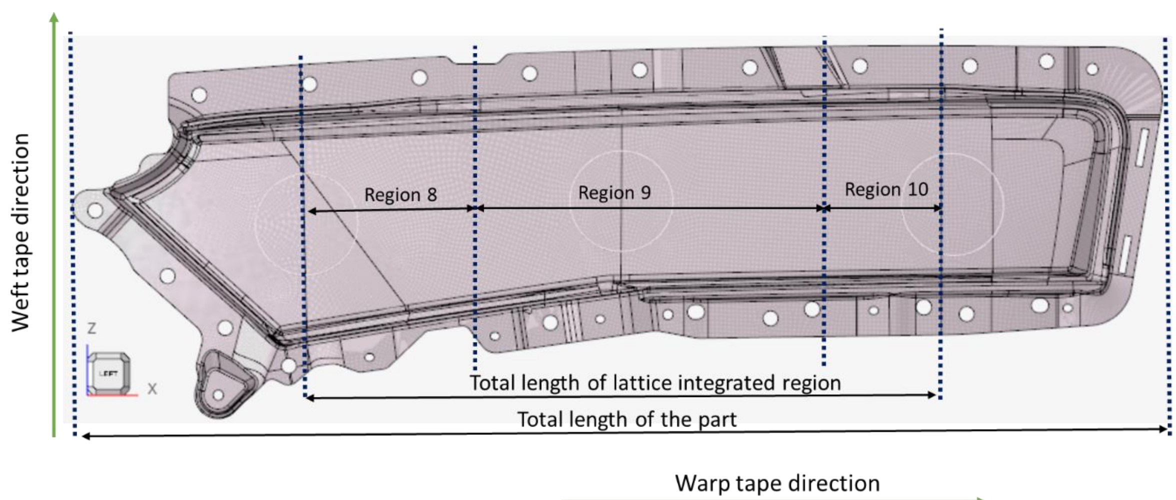


Figure 15 : Door insert sectioned into different regions for heterogenous lattice reinforcement

The final optimized lattice designs that fullfill all the design targets, used to reinforce the NFPP 1200 gsm (Figure 16) and NFPP 1000 gsm (Figure 17) door inserts, are illustrated in the figures below.

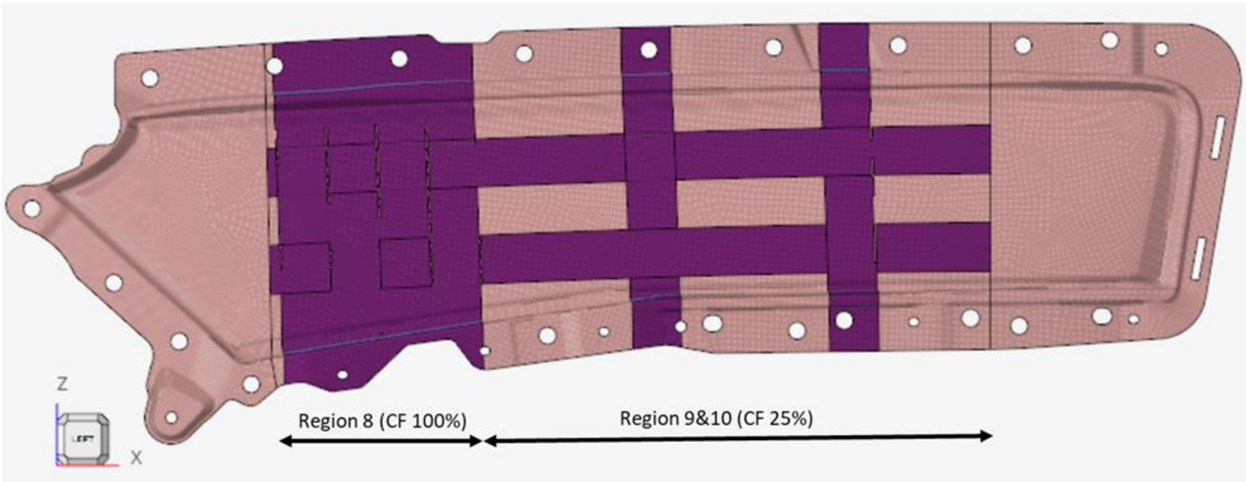


Figure 16 : Optimized lattice Reinforcement for NFPP (1200 gsm)

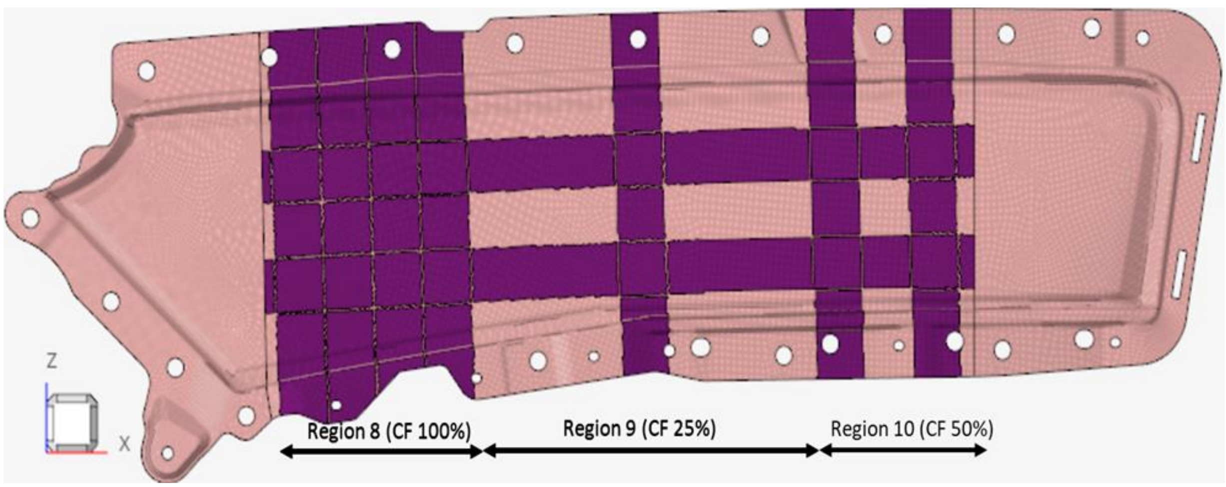


Figure 17 : Optimized lattice Reinforcement for NFPP (1000 gsm)

A summary of iterations for both NFPP design cases is presented below. As shown in Table 6, iteration 3 as lattice reinforcement for NFPP1200 gsm with a weft cover factor of 25% for regions 8 and 9 and 100% for region 10 achieved deflections below the allowable target. Similarly, iteration 2 with weft cover factors of 50%, 25%, and 100% for regions 8, 9, and 10 respectively as lattice reinforcement design for NFPP 100 gsm also produced deflections within the allowable limit. The peak Von Mises stress values in the NFPP material for both final iterations were about 20 MPa which were below the yield strength specified in the NFPP data sheet (38 MPa), confirming that the integration of lattices in the door component successfully meets the target stiffness and strength requirements.

Table 6 : Summary of Iterations for a Final Optimized Lattice Design (marked in green)

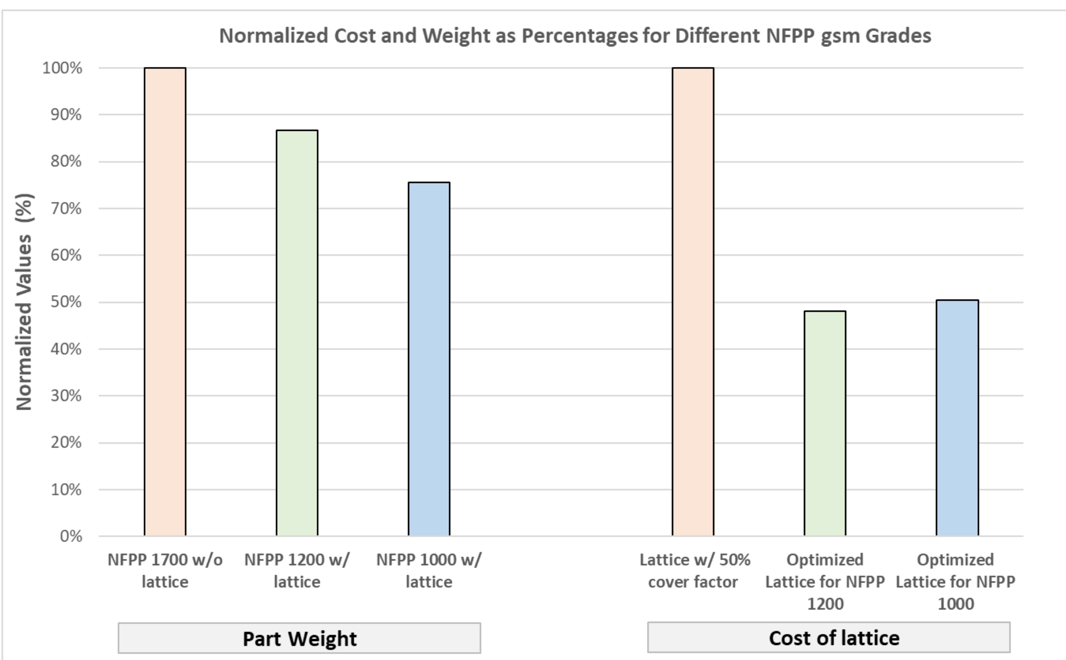
	Iteration	Cover Factor of Weft Tapes			Deflection		
		Region 8	Region 9	Region 10	Region 8	Region 9	Region 10
NFPP 1200 gsm	1	50%	50%	50%	Pass	Pass	Pass
	2	100%	25%	500%	Pass	Pass	Pass
	3	100%	25%	25%	Pass	Pass	Pass
NFPP 1000 gsm	1	50%	50%	50%	Pass	Pass	Pass
	2	100%	25%	50%	Pass	Pass	Pass
	3	100%	25%	25%	Fail	Pass	Pass

6.5. Cost and Weight Optimzation

The integration of lattice reinforcement into Natural Fibre Polypropylene (NFPP) composites has led to substantial weight and cost savings. The original NFPP part, made from 1700 gsm material with a thickness of 1.8 mm, was optimized to reduce the thickness to 1.64 mm and 1.46 mm for the door inserts fabricated from 1200 gsm and 1000 gsm materials, respectively.

Figure 18 presents normalized values of cost and weight as percentages for different NFPP grades (1700, 1200, and 1000 gsm). The optimized lattice design solutions achieved weight savings of 24% for the 1200 gsm version and 13% for the 1000 gsm version, compared to reinforcing the door insert with a lattice design of 50% cover factor. Additionally, cost savings of 52% for the 1200 gsm version and 50% for the 1000 gsm version were achieved when reinforced with the optimized lattice designs, compared to the original unreinforced 1700 gsm door insert.

These results underscore the effectiveness of optimized lattice reinforcement in reducing both material costs and overall weight of NFPP door insert.



.

Figure 18 : Comparison of Baseline and Final Designs w/ and w/o Optimizations

7. CONCLUSION AND FUTURE WORK

In conclusion, this paper has demonstrated a robust methodology for optimizing automotive door components using composite lattice reinforcements integrated through advanced Finite Element Analysis (FEA) techniques. The integration of WEAV3D's Rebar for Plastics® technology with Altair HyperWorks has enabled precise modeling and optimization of hybrid-material structures, overcoming challenges in traditional FEA approaches.

Key contributions include:

1. **FEA Methodology Advancements:** The use of explicit FEA modeling techniques in Altair HyperMesh facilitated accurate representation of complex lattice designs, allowing for parameterization of tape materials, spacing, and layer counts. This approach streamlined design iterations and enabled regional optimization within the automotive door components.
2. **Validation through Experimental Testing:** The paper validated the FEA predictions against experimental data from three-point bend tests, showing good correlation and confirming the reliability of the explicit modeling approach. This validation underscores the efficacy of the methodology in predicting mechanical behavior and guiding design decisions.
3. **Optimization and Performance Enhancement:** Through iterative optimization, heterogeneous lattice designs were tailored to meet specific design targets such as stiffness, strength, weight reduction, and cost efficiency. This approach significantly improved the performance of door inserts made from different material grades of natural fiber polypropylene (NFPP), achieving up to 24% weight savings while maintaining or enhancing mechanical properties.

The Explicit Model approach using Altair HyperWorks described in this paper still has certain limitations that need to be addressed. Each lattice design iteration still requires the user intervention to needs to manually update the input text file with tow and bulk definitions, which needs to be repeated each time until an optimized lattice design solution is achieved that meets the desired performance, cost, and weight targets. Another limitation relates to the scripts ability to handle complex geometries as the script is unable to accommodate sharp radii or significant.

Future work will involve refining the script to accommodate complex part geometries by implementing advanced projection or draping algorithms. These algorithms will improve the accuracy of placing lattice structures on curved or irregular surfaces, which is crucial for optimizing structural performance in automotive and aerospace applications. Additionally, there will be a focus on advancing a multiscale implicit modeling approach within Altair that aims to fully automate the optimization of lattice patterns, allowing for rapid exploration of design alternatives and accurate prediction of average stresses with the bulk material and lattice reinforcement. Once candidate designs are identified through implicit modeling, the explicit model can be used to precisely model these designs, ensuring precise prediction of stress distributions and mechanical behavior. These future enhancements are expected to significantly improve the efficiency and robustness of the FEA methodology, reducing the need for manual intervention, and accelerating the design optimization process for complex geometries in various industrial sectors.

We would also like to thank Antolin for their support throughout the automotive door panel case study. Their expertise was instrumental in shaping the work and showcasing a methodology for adopting composite lattice reinforcements in the automotive sector.

8. REFERENCES

[1] Rochardjo, H.S.B.; Budiyantoro, C. Manufacturing and Analysis of Overmolded Hybrid Fiber Polyamide 6 Composite. *Polymers* (2021):1-3.

[2] Y. A. Gowayed, C. Pastore, and C. S. Howarth, "Modification and application of a unit cell continuum model to predict the elastic properties of textile composites," *Composites Part A (Applied Science and Manufacturing),* vol. 27A, pp. 149-55, / 1996.

[3] T. Ishikawa and T. W. Chou, "One-dimensional micromechanical analysis of woven fabric composites," *AIAA Journal,* vol. 21, pp. 1714-21, 12/ 1983.

[4] Stig, Fredrik & Hallström, Stefan. (2013). Influence of crimp on 3D-woven fibre reinforced composites. Composite Structures. 95. 114–122. 10.1016/j.compstruct.2012.07.022.

[5] M. Kamble and C. Oberste, "A Novel FEA Approach to Design and Optimize Composite Lattice Reinforcements and Simulate the Mechanical properties of Composite Reinforced Plastics." *CAMX Conference Proceedings*. Anaheim, CA, October 17-20, 2022. CAMX – *The Composites and Advanced Materials Expo.*